ME1303 GAS DYNAMICS AND JET PROPULSION

COMPILED BY,

C.SELVAM

ASSISTANT PROFESSOR

DEPARTMENT OF MECHANICAL ENGINEERING

ME602 GAS DYNAMICS AND JET PROPULSION 3 1 0 100

OBJECTIVES

To Understand the basic difference between incompressible and compressible flow
To study the phenomenon of shock waves and its effect on flow
To gain basic knowledge about jet propulsion and Rocket Propulsion

1. COMPRESSIBLE FLOW – FUNDAMENTALS 8

Energy and momentum equations for compressible fluid flows, various regions of flows, reference velocities, stagnation state, velocity of sound, critical states, Mach number, critical Mach number, types of waves, Mach cone, Mach angle, effect of Mach number on compressibility

2. FLOW THROUGH VARIABLE AREA DUCTS 9

Isentropic flow through variable area ducts, T-s and h-s diagrams for nozzle and diffuser flows, area ratio as a function of Mach number, mass flow rate through nozzles and diffusers, effect of friction in flow through nozzles.

3. FLOW THROUGH CONSTANT AREA DUCTS 10

Flow in constant area ducts with friction (Fanno flow) – Fanno curves and Fanno flow equation, variation of flow properties, variation of Mach number with duct length.Isothermal flow with friction in constant area ducts Flow in constant area ducts with heat transfer (Rayleigh flow), Rayleigh line and Rayleigh flow equation, variation of flow properties, maximum heat transfer.

4. NORMAL SHOCK 8

Governing equations, variation of flow parameters like static pressure, static temperature, density, stagnation pressure and entropy across the normal shock, Prandtl - Meyer equation, impossibility of shock in subsonic flows, flow in convergent and divergent nozzle with shock, normal shock in Fanno and Rayleigh flows, flow with oblique shock (elementary treatment only).

5. PROPULSION 10

Aircraft propulsion – types of jet engines – energy flow through jet engines, study of turbojet engine components – diffuser, compressor, combustion chamber, turbine and exhaust systems, performance of turbo jet engines – thrust, thrust power, propulsive and overall efficiencies, thrust augmentation in turbo jet engine, ram jet and pulse jet engines Rocket propulsion – rocket engines thrust equation – effective jet velocity specific impulse – rocket engine performance, solid and liquid propellants, comparison of different propulsion systems.

TUTORIAL 15

TOTAL : 60

Note: *(Use of approved gas tables is permitted in the University examination)*

TEXT BOOKS

1. Yahya. S.M., “Fundamental of compressible flow”, New Age International (p) Ltd., New Delhi, 1996.
2. Patrich.H. Oosthvizen, William E.Carscallen, “Compressible fluid flow”, McGraw-Hill, 1997

REFERENCES

1. Cohen. H., Rogers R.E.C and Sravanamutoo, “Gas turbine theory”, Addison Wesley Ltd., 1987.
2. Ganesan. V., “Gas Turbines”, Tata McGraw-Hill, New Delhi, 1999
3. Rathakrishnan.E, “Gas Dynamics”, Prentice Hall of India, New Delhi, 2001

ME602 GAS DYNAMICS AND JET PROPULSION

UNIT I COMPRESSIBLE FLOW – FUNDAMENTALS

In physics, **fluid dynamics** is a sub-discipline of fluid mechanics that deals with **fluid flow**—the natural science of fluids (liquids and gases) in motion. It has several subdisciplines itself, including aerodynamics (the study of air and other gases in motion) and **hydrodynamics** (the study of liquids in motion). Fluid dynamics has a wide range of applications, including calculating forces and moments on aircraft, determining the mass flow rate of petroleum through pipelines, predicting weather patterns, understanding nebulae in interstellar space and reportedly modeling fission weapon detonation. Some of its principles are even used in traffic engineering, where traffic is treated as a continuous fluid.

Fluid dynamics offers a systematic structure that underlies these practical disciplines, that embraces empirical and semi-empirical laws derived from flow measurement and used to solve practical problems. The solution to a fluid dynamics problem typically involves calculating various properties of the fluid, such as velocity, pressure, density, and temperature, as functions of space and time.

Historically, *hydrodynamics* meant something different than it does today. Before the twentieth century, hydrodynamics was synonymous with fluid dynamics. This is still reflected in names of some fluid dynamics topics, like magnetohydrodynamics and hydrodynamic stability—both also applicable in, as well as being applied to, gases.a

The foundational axioms of fluid dynamics are the conservation laws, specifically, conservation of mass, conservation of linear momentum (also known as Newton's Second Law of Motion), and conservation of energy (also known as First Law of Thermodynamics). These are based on classical mechanics and are modified in quantum mechanics and general relativity. They are expressed using the Reynolds Transport Theorem.

In addition to the above, fluids are assumed to obey the *continuum assumption*. Fluids are composed of molecules that collide with one another and solid objects. However, the continuum assumption considers fluids to be continuous, rather than discrete. Consequently, properties such

as density, pressure, temperature, and velocity are taken to be well-defined at infinitesimally small points, and are assumed to vary continuously from one point to another. The fact that the fluid is made up of discrete molecules is ignored.

For fluids which are sufficiently dense to be a continuum, do not contain ionized species, and have velocities small in relation to the speed of light, the momentum equations for Newtonian fluids are the Navier-Stokes equations, which is a non-linear set of differential equations that describes the flow of a fluid whose stress depends linearly on velocity gradients and pressure. The unsimplified equations do not have a general closed-form solution, so they are primarily of use in Computational Fluid Dynamics. The equations can be simplified in a number of ways, all of which make them easier to solve. Some of them allow appropriate fluid dynamics problems to be solved in closed form.

In addition to the mass, momentum, and energy conservation equations, a thermodynamical equation of state giving the pressure as a function of other thermodynamic variables for the fluid is required to completely specify the problem. An example of this would be the perfect gas equation of state:

where p is pressure, ρ is density, R_u is the gas constant, M is the molar mass and T is temperature.

Compressible vs incompressible flow

All fluids are compressible to some extent, that is changes in pressure or temperature will result in changes in density. However, in many situations the changes in pressure and temperature are sufficiently small that the changes in density are negligible. In this case the flow can be modeled as an incompressible flow. Otherwise the more general compressible flow equations must be used.

Mathematically, incompressibility is expressed by saying that the density ρ of a fluid parcel does not change as it moves in the flow field, i.e.,

where D / D*t* is the substantial derivative, which is the sum of local and convective derivatives. This additional constraint simplifies the governing equations, especially in the case when the fluid has a uniform density.

For flow of gases, to determine whether to use compressible or incompressible fluid dynamics, the Mach number of the flow is to be evaluated. As a rough guide, compressible effects can be ignored at Mach numbers below approximately 0.3. For liquids, whether the incompressible assumption is valid depends on the fluid properties (specifically the critical pressure and temperature of the fluid) and the flow conditions (how close to the critical pressure the actual flow pressure becomes). Acoustic problems always require allowing compressibility, since sound waves are compression waves involving changes in pressure and density of the medium through which they propagate.

viscous vs inviscid flow

Viscous problems are those in which fluid friction has significant effects on the fluid motion.

The Reynolds number, which is a ratio between inertial and viscous forces, can be used to evaluate whether viscous or inviscid equations are appropriate to the problem.

Stokes flow is flow at very low Reynolds numbers, $Re \ll 1$, such that inertial forces can be neglected compared to viscous forces.

On the contrary, high Reynolds numbers indicate that the inertial forces are more significant than the viscous (friction) forces. Therefore, we may assume the flow to be an inviscid flow, an approximation in which we neglect viscosity completely, compared to inertial terms.

This idea can work fairly well when the Reynolds number is high. However, certain problems such as those involving solid boundaries, may require that the viscosity be included. Viscosity often cannot be neglected near solid boundaries because the no-slip condition can generate a thin region of large strain rate (known as Boundary layer) which enhances the effect of even a small amount of viscosity, and thus generating vorticity. Therefore, to calculate net forces on bodies (such as wings) we should use viscous flow equations. As illustrated by d'Alembert's paradox, a body in an inviscid fluid will experience no drag force. The standard equations of inviscid flow are the Euler equations. Another often used model, especially in computational fluid dynamics, is to use the Euler equations away from the body and the boundary layer equations, which incorporates viscosity, in a region close to the body.

The Euler equations can be integrated along a streamline to get Bernoulli's equation. When the flow is everywhere irrotational and inviscid, Bernoulli's equation can be used throughout the flow field. Such flows are called potential flows.

Steady vs unsteady flow

When all the time derivatives of a flow field vanish, the flow is considered to be a **steady flow**. Steady-state flow refers to the condition where the fluid properties at a point in the system do not change over time. Otherwise, flow is called unsteady. Whether a particular flow is steady or unsteady, can depend on the chosen frame of reference. For instance, laminar flow over a sphere is steady in the frame of reference that is stationary with respect to the sphere. In a frame of reference that is stationary with respect to a background flow, the flow is unsteady.

Turbulent flows are unsteady by definition. A turbulent flow can, however, be statistically stationary.

The random field $U(x,t)$ is statistically stationary if all statistics are invariant under a shift in time.

This roughly means that all statistical properties are constant in time. Often, the mean field is the object of interest, and this is constant too in a statistically stationary flow.

Steady flows are often more tractable than otherwise similar unsteady flows. The governing equations of a steady problem have one dimension fewer (time) than the governing equations of the same problem without taking advantage of the steadiness of the flow field.

Laminar vs turbulent flow

Turbulence is flow characterized by recirculation, eddies, and apparent randomness. Flow in which turbulence is not exhibited is called laminar. It should be noted, however, that the presence of eddies or recirculation alone does not necessarily indicate turbulent flow—these phenomena may be present in laminar flow as well. Mathematically, turbulent flow is often represented via a Reynolds decomposition, in which the flow is broken down into the sum of an average component and a perturbation component.

It is believed that turbulent flows can be described well through the use of the Navier–Stokes equations. Direct numerical simulation (DNS), based on the Navier–Stokes equations, makes it possible to simulate turbulent flows at moderate Reynolds numbers. Restrictions depend on the power of the computer used and the efficiency of the solution algorithm. The results of DNS have been found to agree well with experimental data for some flows[4].

Most flows of interest have Reynolds numbers much too high for DNS to be a viable option[5], given the state of computational power for the next few decades. Any flight vehicle large enough to carry a human (L > 3 m), moving faster than 72 km/h (20 m/s) is well beyond the limit of DNS simulation (Re = 4 million). Transport aircraft wings (such as on an Airbus A300 or Boeing 747) have Reynolds numbers of 40 million (based on the wing chord). In order to solve these real-life flow problems, turbulence models will be a necessity for the foreseeable future. Reynolds-averaged Navier–Stokes equations (RANS) combined with turbulence modeling provides a model of the effects of the turbulent flow. Such a modeling mainly provides the additional momentum transfer by the Reynolds stresses, although the turbulence also enhances the heat and mass transfer. Another promising methodology is large eddy simulation (LES), especially in the guise of detached eddy simulation (DES)—which is a combination of RANS turbulence modeling and large eddy simulation.

Newtonian vs non-Newtonian fluids

Sir Isaac Newton showed how stress and the rate of strain are very close to linearly related for many familiar fluids, such as water and air. These Newtonian fluids are modeled by a coefficient called viscosity, which depends on the specific fluid.

However, some of the other materials, such as emulsions and slurries and some visco-elastic materials (e.g. blood, some polymers), have more complicated *non-Newtonian* stress-strain behaviours. These materials include *sticky liquids* such as latex, honey, and lubricants which are studied in the sub-discipline of rheology.

Subsonic vs transonic, supersonic and hypersonic flows

While many terrestrial flows (e.g. flow of water through a pipe) occur at low mach numbers, many flows of practical interest (e.g. in aerodynamics) occur at high fractions of the Mach Number M=1 or in excess of it (supersonic flows). New phenomena occur at these Mach number regimes (e.g. shock waves for supersonic flow, transonic instability in a regime of flows with M nearly equal to 1, non-equilibrium chemical behavior due to ionization in hypersonic flows) and it is necessary to treat each of these flow regimes separately.

Magnetohydrodynamics

Magnetohydrodynamics is the multi-disciplinary study of the flow of electrically conducting fluids in electromagnetic fields. Examples of such fluids include plasmas, liquid metals, and salt water. The fluid flow equations are solved simultaneously with Maxwell's equations of electromagnetism.

Other approximations

There are a large number of other possible approximations to fluid dynamic problems. Some of the more commonly used are listed below.

- The **Boussinesq approximation** neglects variations in density except to calculate buoyancy forces. It is often used in free convection problems where density changes are small.
- **Lubrication theory** and **Hele-Shaw flow** exploits the large aspect ratio of the domain to show that certain terms in the equations are small and so can be neglected.
- **Slender-body theory** is a methodology used in Stokes flow problems to estimate the force on, or flow field around, a long slender object in a viscous fluid.
- The **shallow-water equations** can be used to describe a layer of relatively inviscid fluid with a free surface, in which surface gradients are small.
- The **Boussinesq equations** are applicable to surface waves on thicker layers of fluid and with steeper surface slopes.

- **Darcy's law** is used for flow in porous media, and works with variables averaged over several pore-widths.
- In rotating systems, the **quasi-geostrophic approximation** assumes an almost perfect balance between pressure gradients and the Coriolis force. It is useful in the study of atmospheric dynamics.

Terminology in incompressible fluid dynamics

The concepts of total pressure and dynamic pressure arise from Bernoulli's equation and are significant in the study of all fluid flows. (These two pressures are not pressures in the usual sense—they cannot be measured using an aneroid, Bourdon tube or mercury column.) To avoid potential ambiguity when referring to pressure in fluid dynamics, many authors use the term static pressure to distinguish it from total pressure and dynamic pressure. Static pressure is identical to pressure and can be identified for every point in a fluid flow field.

In *Aerodynamics*, L.J. Clancy writes[6]: *To distinguish it from the total and dynamic pressures, the actual pressure of the fluid, which is associated not with its motion but with its state, is often referred to as the static pressure, but where the term pressure alone is used it refers to this static pressure.*

A point in a fluid flow where the flow has come to rest (i.e. speed is equal to zero adjacent to some solid body immersed in the fluid flow) is of special significance. It is of such importance that it is given a special name—a stagnation point. The static pressure at the stagnation point is of special significance and is given its own name—stagnation pressure. In incompressible flows, the stagnation pressure at a stagnation point is equal to the total pressure throughout the flow field.

Terminology in compressible fluid dynamics

In a compressible fluid, such as air, the temperature and density are essential when determining the state of the fluid. In addition to the concept of total pressure (also known as stagnation pressure), the concepts of total (or stagnation) temperature and total (or stagnation) density are also essential in any study of compressible fluid flows. To avoid potential ambiguity when

referring to temperature and density, many authors use the terms static temperature and static density. Static temperature is identical to temperature; and static density is identical to density; and both can be identified for every point in a fluid flow field.

The temperature and density at a stagnation point are called stagnation temperature and stagnation density.

A similar approach is also taken with the thermodynamic properties of compressible fluids. Many authors use the terms total (or stagnation) enthalpy and total (or stagnation) entropy. The terms static enthalpy and static entropy appear to be less common, but where they are used they mean nothing more than enthalpy and entropy respectively, and the prefix "static" is being used to avoid ambiguity with their 'total' or 'stagnation' counterparts. Because the 'total' flow conditions are defined by isentropically bringing the fluid to rest, the total (or stagnation) entropy is by definition always equal to the "static" entropy.

The Mach number is commonly used both with objects traveling at high speed in a fluid, and with high-speed fluid flows inside channels such as nozzles, diffusers or wind tunnels. As it is defined as a ratio of two speeds, it is a dimensionless number. At Standard Sea Level conditions (corresponding to a temperature of 15 degrees Celsius), the speed of sound is 340.3 m/s[3] (1225 km/h, or 761.2 mph, or 661.5 knots, or 1116 ft/s) in the Earth's atmosphere. The speed represented by Mach 1 is not a constant; for example, it is mostly dependent on temperature and atmospheric composition and largely independent of pressure. In the stratosphere, where the temperatures are constant, it does not vary with altitude even though the air pressure changes significantly with altitude.

Since the speed of sound increases as the temperature increases, the actual speed of an object traveling at Mach 1 will depend on the fluid temperature around it. Mach number is useful because the fluid behaves in a similar way at the same Mach number. So, an aircraft traveling at Mach 1 at 20°C or 68°F will experience shock waves in much the same manner as when it is traveling at Mach 1 at 11,000 m (36,000 ft) at -50°C or -58F, even though it is traveling at only 86% of its speed at higher temperature like 20°C or 68°F.

High-speed flow around objects

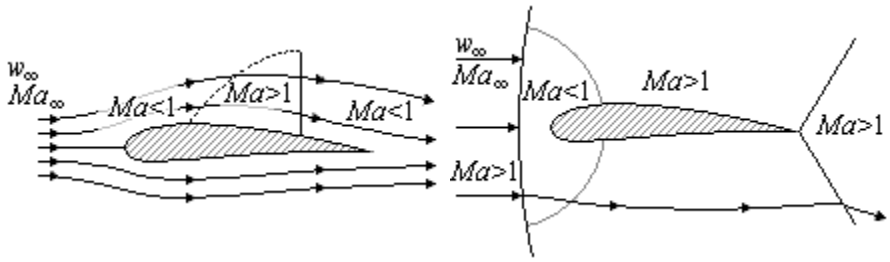
Flight can be roughly classified in six categories:

Regime	**Subsonic**	**Transonic**	**Sonic**	**Supersonic**	**Hypersonic**	**High-hypersonic**
Mach	<0.75	0.75–1.2	1.0	1.2–5.0	5.0–10.0	>10.0

For comparison: the required speed for low Earth orbit is approximately 7.5 km/s = Mach 25.4 in air at high altitudes. The speed of light in a vacuum corresponds to a Mach number of approximately 881,000 (relative to air at sea level).

At transonic speeds, the flow field around the object includes both sub- and supersonic parts. The transonic period begins when first zones of M>1 flow appear around the object. In case of an airfoil (such as an aircraft's wing), this typically happens above the wing. Supersonic flow can decelerate back to subsonic only in a normal shock; this typically happens before the trailing edge. (Fig.1a)

As the speed increases, the zone of $M>1$ flow increases towards both leading and trailing edges. As $M=1$ is reached and passed, the normal shock reaches the trailing edge and becomes a weak oblique shock: the flow decelerates over the shock, but remains supersonic. A normal shock is created ahead of the object, and the only subsonic zone in the flow field is a small area around the object's leading edge. (Fig.1b)



(a) (b)

Fig. 1. *Mach number in transonic airflow around an airfoil; M<1 (a) and M>1 (b).*

When an aircraft exceeds Mach 1 (i.e. the sound barrier) a large pressure difference is created just in front of the aircraft. This abrupt pressure difference, called a shock wave, spreads backward and outward from the aircraft in a cone shape (a so-called Mach cone). It is this shock

wave that causes the sonic boom heard as a fast moving aircraft travels overhead. A person inside the aircraft will not hear this. The higher the speed, the more narrow the cone; at just over M=1 it is hardly a cone at all, but closer to a slightly concave plane.

At fully supersonic speed, the shock wave starts to take its cone shape and flow is either completely supersonic, or (in case of a blunt object), only a very small subsonic flow area remains between the object's nose and the shock wave it creates ahead of itself. (In the case of a sharp object, there is no air between the nose and the shock wave: the shock wave starts from the nose.)

As the Mach number increases, so does the strength of the shock wave and the Mach cone becomes increasingly narrow. As the fluid flow crosses the shock wave, its speed is reduced and temperature, pressure, and density increase. The stronger the shock, the greater the changes. At high enough Mach numbers the temperature increases so much over the shock that ionization and dissociation of gas molecules behind the shock wave begin. Such flows are called hypersonic.

It is clear that any object traveling at hypersonic speeds will likewise be exposed to the same extreme temperatures as the gas behind the nose shock wave, and hence choice of heat-resistant materials becomes important.

High-speed flow in a channel

As a flow in a channel crosses M=1 becomes supersonic, one significant change takes place. The conservation of mass flow rate leads one to expect that contracting the flow channel would increase the flow speed (i.e. making the channel narrower results in faster air flow) and at subsonic speeds this holds true. However, once the flow becomes supersonic, the relationship of flow area and speed is reversed: expanding the channel actually increases the speed.

The obvious result is that in order to accelerate a flow to supersonic, one needs a convergent-divergent nozzle, where the converging section accelerates the flow to M=1, sonic speeds, and the diverging section continues the acceleration. Such nozzles are called de Laval nozzles and in extreme cases they are able to reach incredible, hypersonic speeds (Mach 13 at 20°C).

An aircraft Machmeter or electronic flight information system (EFIS) can display Mach number derived from stagnation pressure (pitot tube) and static pressure.

Critical Mach number

In aerodynamics, the **critical Mach number (Mcr)** of an aircraft is the lowest Mach number at which the airflow over a small region of the wing reaches the speed of sound.[1]

For all aircraft in flight, the airflow around the aircraft is not exactly the same as the airspeed of the aircraft due to the airflow speeding up and slowing down to travel around the aircraft structure. At the Critical Mach number, local airflow in some areas near the airframe reaches the speed of sound, even though the aircraft itself has an airspeed lower than Mach 1.0. This creates a weak shock wave. At speeds faster than the Critical Mach number:

- drag coefficient increases suddenly, causing dramatically increased drag
- in aircraft not designed for transonic or supersonic speeds, changes to the airflow over the flight control surfaces lead to deterioration in control of the aircraft.

In aircraft not designed to fly at the Critical Mach number, shock waves in the flow over the wing and tailplane were sufficient to stall the wing, make control surfaces ineffective or lead to loss of control such as Mach tuck. The phenomena associated with problems at the Critical Mach number became known as compressibility. Compressibility led to a number of accidents involving high-speed military and experimental aircraft in the 1930s and 1940s.

Although unknown at the time, compressibility was the cause of the phenomenon known as the sound barrier. Subsonic aircraft such as the Supermarine Spitfire, BF 109, P-51 Mustang, Gloster Meteor, Me 262, P-80 have relatively thick, unswept wings and are incapable of reaching Mach 1.0. In 1947, Chuck Yeager flew the Bell X-1 to Mach 1.0 and beyond, and the sound barrier was finally broken.

Early transonic military aircraft such as the Hawker Hunter and F-86 Sabre were designed to fly satisfactorily faster than their Critical Mach number. They did not possess sufficient engine thrust to reach Mach 1.0 in level flight but could be dived to Mach 1.0 and beyond, and remain

controllable. Modern passenger-carrying jet aircraft such as Airbus and Boeing aircraft have Maximum Operating Mach numbers slower than Mach 1.0.

Supersonic aircraft, such as Concorde, the English Electric Lightning, Lockheed F-104, Dassault Mirage III, and MiG 21 are designed to exceed Mach 1.0 in level flight. They have very thin wings. Their Critical Mach numbers are higher than those of subsonic and transonic aircraft but less than Mach 1.0.

The actual Critical Mach number varies from wing to wing. In general a thicker wing will have a lower Critical Mach number, because a thicker wing accelerates the airflow to a faster speed than a thinner one. For instance, the fairly thick wing on the P-38 Lightning led to a Critical Mach number of about .69, a speed it could reach with some ease in dives, which led to a number of crashes. The much thinner wing on the Supermarine Spitfire caused this aircraft to have a Critical Mach number of about 0.89

Effects of Mach number and compressibility

We study the effects of Mach number and compressibility on strain-rate and vorticity dynamics in decaying isotropic turbulence employing direct numerical simulations. Since local Mach number and dilatation are two direct indicators of compressibility of a fluid element, we use these quantities as conditioning parameters to examine the various aspects of turbulence dynamics. Several interesting observations along with the underlying physics pertaining to the inertial (vortex stretching and self-straining) and pressure (pressure Hessian and baroclinic) terms in the budget of strain-rate and vorticity dynamics will be presented in the talk. The contrasting nature of these physical effects in expanding vs. contracting and supersonic vs. subsonic fluid elements will be highlighted.

UNIT-II & III FLOW THROUGH CONSTANT & VARIABLE AREA DUCTS

Rayleigh Flow:

Rayleigh flow refers to diabatic flow through a constant area duct where the effect of heat addition or rejection is considered. Compressibility effects often come into consideration, although the Rayleigh flow model certainly also applies to incompressible flow. For this model,

the duct area remains constant and no mass is added within the duct. Therefore, unlike Fanno flow, the stagnation temperature is a variable. The heat addition causes a decrease in stagnation pressure, which is known as the Rayleigh effect and is critical in the design of combustion systems. Heat addition will cause both supersonic and subsonic Mach numbers to approach Mach 1, resulting in choked flow. Conversely, heat rejection decreases a subsonic Mach number and increases a supersonic Mach number along the duct. It can be shown that for calorically perfect flows the maximum entropy occurs at M = 1. Rayleigh flow is named after John Strutt, 3rd Baron Rayleigh.

Fanno Flow:

Fanno flow refers to adiabatic through a constant area duct where the effect of friction is considered. Compressibilityflow effects often come into consideration, although the Fanno flow model certainly also applies to incompressible flow. For this model, the duct area remains constant, the flow is assumed to be steady and one-dimensional, and no mass is added within the duct. The Fanno flow model is considered an irreversible process due to viscous effects. The viscous friction causes the flow properties to change along the duct. The frictional effect is modeled as a shear stress at the wall acting on the fluid with uniform properties over any cross section of the duct.

For a flow with an upstream Mach number greater than 1.0 in a sufficiently long enough duct, deceleration occurs and the flow can become choked. On the other hand, for a flow with an upstream Mach number less than 1.0, acceleration occurs and the flow can become choked in a sufficiently long duct. It can be shown that for flow of calorically per The Fanno flow model begins with a differential equation that relates the change in Mach number with respect to the length of the duct, dM/dx. Other terms in the differential equation are the heat capacity ratio, γ, the Fanning friction factor, f, and the hydraulic diameter, D_h:

Variation of Fluid Properties:

Equations of fluid dynamics

The foundational axioms of fluid dynamics are the conservation laws, specifically, conservation of mass, conservation of linear momentum (also known as Newton's Second Law of Motion), and conservation of energy (also known as First Law of Thermodynamics). These are based on

classical mechanics and are modified in quantum mechanics and general relativity. They are expressed using the Reynolds Transport Theorem.

In addition to the above, fluids are assumed to obey the *continuum assumption*. Fluids are composed of molecules that collide with one another and solid objects. However, the continuum assumption considers fluids to be continuous, rather than discrete. Consequently, properties such as density, pressure, temperature, and velocity are taken to be well-defined at infinitesimally small points, and are assumed to vary continuously from one point to another. The fact that the fluid is made up of discrete molecules is ignored.

For fluids which are sufficiently dense to be a continuum, do not contain ionized species, and have velocities small in relation to the speed of light, the momentum equations for Newtonian fluids are the Navier-Stokes equations, which is a non-linear set of differential equations that describes the flow of a fluid whose stress depends linearly on velocity gradients and pressure. The unsimplified equations do not have a general closed-form solution, so they are primarily of use in Computational Fluid Dynamics. The equations can be simplified in a number of ways, all of which make them easier to solve. Some of them allow appropriate fluid dynamics problems to be solved in closed form.

In addition to the mass, momentum, and energy conservation equations, a thermodynamical equation of state giving the pressure as a function of other thermodynamic variables for the fluid is required to completely specify the problem. An example of this would be the perfect gas equation of state:

$$p = \frac{\rho R_u T}{M}$$

where p is pressure, ρ is density, R_u is the gas constant, M is the molar mass and T is temperature.

Compressible vs incompressible flow

All fluids are compressible to some extent, that is changes in pressure or temperature will result in changes in density. However, in many situations the changes in pressure and temperature are sufficiently small that the changes in density are negligible. In this case the flow can be modeled

as an incompressible flow. Otherwise the more general compressible flow equations must be used.

Mathematically, incompressibility is expressed by saying that the density ρ of a fluid parcel does not change as it moves in the flow field, i.e.,

$$\frac{\mathrm{D}\rho}{\mathrm{D}t} = 0\,,$$

where D / D*t* is the substantial derivative, which is the sum of local and convective derivatives. This additional constraint simplifies the governing equations, especially in the case when the fluid has a uniform density.

For flow of gases, to determine whether to use compressible or incompressible fluid dynamics, the Mach number of the flow is to be evaluated. As a rough guide, compressible effects can be ignored at Mach numbers below approximately 0.3. For liquids, whether the incompressible assumption is valid depends on the fluid properties (specifically the critical pressure and temperature of the fluid) and the flow conditions (how close to the critical pressure the actual flow pressure becomes). Acoustic problems always require allowing compressibility, since sound waves are compression waves involving changes in pressure and density of the medium through which they propagate.

Viscous vs inviscid flow

Viscous problems are those in which fluid friction has significant effects on the fluid motion.

The Reynolds number, which is a ratio between inertial and viscous forces, can be used to evaluate whether viscous or inviscid equations are appropriate to the problem.

Stokes flow is flow at very low Reynolds numbers, $Re \ll 1$, such that inertial forces can be neglected compared to viscous forces.

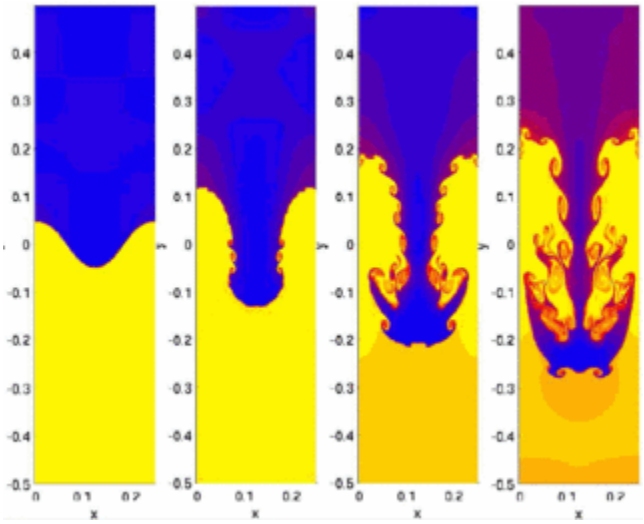
On the contrary, high Reynolds numbers indicate that the inertial forces are more significant than the viscous (friction) forces. Therefore, we may assume the flow to be an inviscid flow, an approximation in which we neglect viscosity completely, compared to inertial terms.

This idea can work fairly well when the Reynolds number is high. However, certain problems such as those involving solid boundaries, may require that the viscosity be included. Viscosity often cannot be neglected near solid boundaries because the no-slip condition can generate a thin region of large strain rate (known as Boundary layer) which enhances the effect of even a small amount of viscosity, and thus generating vorticity. Therefore, to calculate net forces on bodies (such as wings) we should use viscous flow equations. As illustrated by d'Alembert's paradox, a body in an inviscid fluid will experience no drag force. The standard equations of inviscid flow are the Euler equations. Another often used model, especially in computational fluid dynamics, is to use the Euler equations away from the body and the boundary layer equations, which incorporates viscosity, in a region close to the body.

The Euler equations can be integrated along a streamline to get Bernoulli's equation. When the flow is everywhere irrotational and inviscid, Bernoulli's equation can be used throughout the flow field. Such flows are called potential flows.

Steady vs unsteady flow

Hydrodynamics simulation of the Rayleigh–Taylor instability



When all the time derivatives of a flow field vanish, the flow is considered to be a **steady flow**. Steady-state flow refers to the condition where the fluid properties at a point in the system do not change over time. Otherwise, flow is called unsteady. Whether a particular flow is steady or unsteady, can depend on the chosen frame of reference. For instance, laminar flow over a sphere is steady in the frame of reference that is stationary with respect to the sphere. In a frame of reference that is stationary with respect to a background flow, the flow is unsteady.

Turbulent flows are unsteady by definition. A turbulent flow can, however, be statistically stationary. According to Pope

The random field $U(x,t)$ is statistically stationary if all statistics are invariant under a shift in time.

This roughly means that all statistical properties are constant in time. Often, the mean field is the object of interest, and this is constant too in a statistically stationary flow.

Steady flows are often more tractable than otherwise similar unsteady flows. The governing equations of a steady problem have one dimension fewer (time) than the governing equations of the same problem without taking advantage of the steadiness of the flow field.

Laminar vs turbulent flow

Turbulence is flow characterized by recirculation, eddies, and apparent randomness. Flow in which turbulence is not exhibited is called laminar. It should be noted, however, that the presence of eddies or recirculation alone does not necessarily indicate turbulent flow—these phenomena may be present in laminar flow as well. Mathematically, turbulent flow is often represented via a Reynolds decomposition, in which the flow is broken down into the sum of an average component and a perturbation component.

It is believed that turbulent flows can be described well through the use of the Navier–Stokes equations. Direct numerical simulation (DNS), based on the Navier–Stokes equations, makes it

possible to simulate turbulent flows at moderate Reynolds numbers. Restrictions depend on the power of the computer used and the efficiency of the solution algorithm. The results of DNS have been found to agree well with experimental data for some flows.

Most flows of interest have Reynolds numbers much too high for DNS to be a viable option, given the state of computational power for the next few decades. Any flight vehicle large enough to carry a human (L > 3 m), moving faster than 72 km/h (20 m/s) is well beyond the limit of DNS simulation (Re = 4 million). Transport aircraft wings (such as on an Airbus A300 or Boeing 747) have Reynolds numbers of 40 million (based on the wing chord). In order to solve these real-life flow problems, turbulence models will be a necessity for the foreseeable future. Reynolds-averaged Navier–Stokes equations (RANS) combined with turbulence modeling provides a model of the effects of the turbulent flow. Such a modeling mainly provides the additional momentum transfer by the Reynolds stresses, although the turbulence also enhances the heat and mass transfer. Another promising methodology is large eddy simulation (LES), especially in the guise of detached eddy simulation (DES)—which is a combination of RANS turbulence modeling and large eddy simulation.

Newtonian vs non-Newtonian fluids

Sir Isaac Newton showed how stress and the rate of strain are very close to linearly related for many familiar fluids, such as water and air. These Newtonian fluids are modeled by a coefficient called viscosity, which depends on the specific fluid.

However, some of the other materials, such as emulsions and slurries and some visco-elastic materials (e.g. blood, some polymers), have more complicated *non-Newtonian* stress-strain behaviours. These materials include *sticky liquids* such as latex, honey, and lubricants which are studied in the sub-discipline of rheology.

Subsonic vs transonic, supersonic and hypersonic flows

While many terrestrial flows (e.g. flow of water through a pipe) occur at low mach numbers, many flows of practical interest (e.g. in aerodynamics) occur at high fractions of the Mach Number M=1 or in excess of it (supersonic flows). New phenomena occur at these Mach number regimes (e.g. shock waves for supersonic flow, transonic instability in a regime of flows with M nearly equal to 1, non-equilibrium chemical behavior due to ionization in hypersonic flows) and it is necessary to treat each of these flow regimes separately.

Magnetohydrodynamics

Magnetohydrodynamics is the multi-disciplinary study of the flow of electrically conducting fluids in electromagnetic fields. Examples of such fluids include plasmas, liquid metals, and salt water. The fluid flow equations are solved simultaneously with Maxwell's equations of electromagnetism.

Use of Tables and Charts:
Fanno Flow:

Fanno flow refers to adiabatic flow through a constant area duct where the effect **...** The equation above can be used to plot the Fanno line

Rayleigh flow

Rayleigh flow refers to adiabatic flow through a constant area duct where the effect **...** Therefore, unlike Fanno flow , the stagnation

UNIT IV NORMAL SHOCK

A **shock wave** (also called **shock front** or simply "**shock**") is a type of propagating disturbance. Like an ordinary wave, it carries energy and can propagate through a medium (solid, liquid, gas or plasma) or in some cases in the absence of a material medium, through a field such as the electromagnetic field. Shock waves are characterized by an abrupt, nearly discontinuous change in the characteristics of the medium.[1] Across a shock there is always an extremely rapid rise in pressure, temperature and density of the flow. In supersonic flows, expansion is achieved through an expansion fan. A shock wave travels through most media at a higher speed than an ordinary wave.

Unlike solitons (another kind of nonlinear wave), the energy of a shock wave dissipates relatively quickly with distance. Also, the accompanying expansion wave approaches and eventually merges with the shock wave, partially cancelling it out. Thus the sonic boom associated with the passage of a supersonic aircraft is the sound wave resulting from the degradation and merging of the shock wave and the expansion wave produced by the aircraft.

When a shock wave passes through matter, the total energy is preserved but the energy which can be extracted as work decreases and entropy increases. This, for example, creates additional drag force on aircraft with shocks.

Shock waves can be:

- Normal: at 90° (perpendicular) to the shock medium's flow direction.
- Oblique: at an angle to the direction of flow.
- Bow: Occurs upstream of the front (bow) of a blunt object when the upstream velocity exceeds Mach 1.

Some other terms

- Shock Front: an alternative name for the shock wave itself
- Contact Front: in a shock wave caused by a driver gas (for example the "impact" of a high explosive on the surrounding air), the boundary between the driver (explosive products) and the driven (air) gases. The Contact Front trails the Shock Front.

In supersonic flows

Pressure-time diagram at an external observation point for the case of a supersonic object propagating past the observer. The leading edge of the object causes a shock (left, in red) and the trailing edge of the object causes an expansion (right, in blue).

When an object (or disturbance) moves faster than the information about it can be propagated into the surrounding fluid, fluid near the disturbance cannot react or "get out of the way" before the disturbance arrives. In a shock wave the properties of the fluid (density, pressure, temperature, velocity, Mach number) change almost instantaneously. Measurements of the

thickness of shock waves have resulted in values approximately one order of magnitude greater than the mean free path of the gas investigated.

Shock waves form when the speed of a gas changes by more than the speed of sound.[2] At the region where this occurs sound waves traveling against the flow reach a point where they cannot travel any further upstream and the pressure progressively builds in that region, and a high pressure shock wave rapidly forms.

Shock waves are not conventional sound waves; a shock wave takes the form of a very sharp change in the gas properties on the order of a few mean free paths (roughly micro-meters at atmospheric conditions) in thickness. Shock waves in air are heard as a loud "crack" or "snap" noise. Over longer distances a shock wave can change from a nonlinear wave into a linear wave, degenerating into a conventional sound wave as it heats the air and loses energy. The sound wave is heard as the familiar "thud" or "thump" of a sonic boom, commonly created by the supersonic flight of aircraft.

The shock wave is one of several different ways in which a gas in a supersonic flow can be compressed. Some other methods are isentropic compressions, including Prandtl-Meyer compressions. The method of compression of a gas results in different temperatures and densities for a given pressure ratio, which can be analytically calculated for a non-reacting gas. A shock wave compression results in a loss of total pressure, meaning that it is a less efficient method of compressing gases for some purposes, for instance in the intake of a scramjet. The appearance of pressure-drag on supersonic aircraft is mostly due to the effect of shock compression on the flow.

Due to nonlinear steepening

Shock waves can form due to steepening of ordinary waves. The best-known example of this phenomenon is ocean waves that form breakers on the shore. In shallow water, the speed of surface waves is dependent on the depth of the water. An incoming ocean wave has a slightly higher wave speed near the crest of each wave than near the troughs between waves, because the wave height is not infinitesimal compared to the depth of the water. The crests overtake the troughs until the leading edge of the wave forms a vertical face and spills over to form a turbulent shock (a breaker) that dissipates the wave's energy as sound and heat.

Similar phenomena affect strong sound waves in gas or plasma, due to the dependence of the sound speed on temperature and pressure. Strong waves heat the medium near each pressure front, due to adiabatic compression of the air itself, so that high pressure fronts outrun the corresponding pressure troughs. While shock formation by this process does not normally happen to sound waves in Earth's atmosphere, it is thought to be one mechanism by which the solar chromosphere and corona are heated, via waves that propagate up from the solar interior.

Analogies

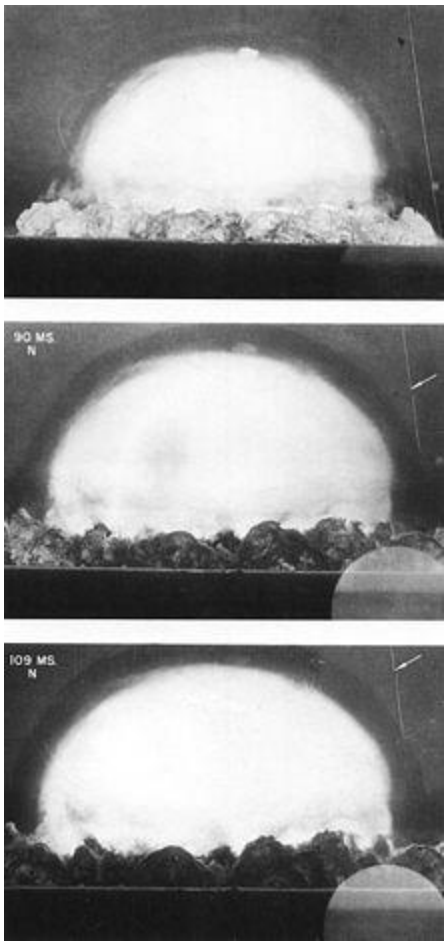
A shock wave may be described as the furthest point upstream of a moving object which "knows" about the approach of the object. In this description, the shock wave position is defined as the boundary between the zone having no information about the shock-driving event, and the zone aware of the shock-driving event, analogous with the light cone described in the theory of special relativity.

To get a shock wave something has to be travelling faster than the local speed of sound. In that case some parts of the air around the aircraft are travelling at exactly the speed of sound with the aircraft, so that the sound waves leaving the aircraft pile up on each other, similar to a tailback on a road, and a shock wave forms, the pressure increases, and then spreads out sideways. Because of this amplification effect, a shock wave is very intense, more like an explosion when heard (not coincidentally, since explosions create shock waves).

Analogous phenomena are known outside fluid mechanics. For example, particles accelerated beyond the speed of light in a refractive medium (where the speed of light is less than that in a vacuum, such as water) create visible shock effects, a phenomenon known as Cherenkov radiation.

Examples

Below are a number of examples of shock waves, broadly grouped with similar shock phenomena:



Shock wave propagating into a stationary medium, ahead of the fireball of an explosion. The shock is made visible by the shadow effect (Trinity explosion.)

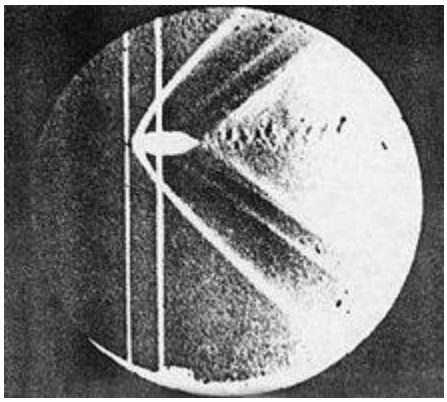
Moving shock

- Usually consists of a shockwave propagating into a stationary medium
- In this case, the gas ahead of the shock is stationary (in the laboratory frame), and the gas behind the shock is supersonic in the laboratory frame. The shock propagates with a wave front which is normal (at right angles) to the direction of flow. The speed of the shock is a function of the original pressure ratio between the two bodies of gas.
- Moving shocks are usually generated by the interaction of two bodies of gas at different pressure, with a shock wave propagating into the lower pressure gas, and an expansion wave propagating into the higher pressure gas.
- Examples: Balloon bursting, Shock tube, shock wave from explosion

Detonation wave

Main article: Detonation

- A detonation wave is essentially a shock supported by a trailing exothermic reaction. It involves a wave traveling through a highly combustible or chemically unstable medium, such as an oxygen-methane mixture or a high explosive. The chemical reaction of the medium occurs following the shock wave, and the chemical energy of the reaction drives the wave forward.
- A detonation wave follows slightly different rules from an ordinary shock since it is driven by the chemical reaction occurring behind the shock wave front. In the simplest theory for detonations, an unsupported, self-propagating detonation wave proceeds at the Chapman-Jouguet velocity. A detonation will also cause a shock of type 1, above to propagate into the surrounding air due to the overpressure induced by the explosion.
- When a shockwave is created by high explosives such as TNT (which has a detonation velocity of 6,900 m/s), it will always travel at high, supersonic velocity from its point of origin.



Shadowgraph of the detached shock on a bullet in supersonic flight, published by Ernst Mach in 1887.

Detached shock

- These shocks are curved, and form a small distance in front of the body. Directly in front of the body, they stand at 90 degrees to the oncoming flow, and then curve around the body. Detached shocks allow the same type of analytic calculations as for the attached

shock, for the flow near the shock. They are a topic of continuing interest, because the rules governing the shock's distance ahead of the blunt body are complicated, and are a function of the body's shape. Additionally, the shock standoff distance varies drastically with the temperature for a non-ideal gas, causing large differences in the heat transfer to the thermal protection system of the vehicle. See the extended discussion on this topic at Atmospheric reentry. These follow the "strong-shock" solutions of the analytic equations, meaning that for some oblique shocks very close to the deflection angle limit, the downstream Mach number is subsonic. See also bow shock or oblique shock

- Such a shock occurs when the maximum deflection angle is exceeded. A detached shock is commonly seen on blunt bodies, but may also be seen on sharp bodies at low Mach numbers.
- Examples: Space return vehicles (Apollo, Space shuttle), bullets, the boundary (Bow shock) of a magnetosphere. The name "bow shock" comes from the example of a bow wave, the detached shock formed at the bow (front) of a ship or boat moving through water, whose slow surface wave speed is easily exceeded (see ocean surface wave).

Attached shock

- These shocks appear as "attached" to the tip of a sharp body moving at supersonic speeds.
- Examples: Supersonic wedges and cones with small apex angles
- The attached shock wave is a classic structure in aerodynamics because, for a perfect gas and inviscid flow field, an analytic solution is available, such that the pressure ratio, temperature ratio, angle of the wedge and the downstream Mach number can all be calculated knowing the upstream Mach number and the shock angle. Smaller shock angles are associated with higher upstream Mach numbers, and the special case where the shock wave is at 90 degrees to the oncoming flow (Normal shock), is associated with a Mach number of one. These follow the "weak-shock" solutions of the analytic equations.

Recompression shock

- These shocks appear when the flow over a transonic body is decelerated to subsonic speeds.
- Examples: Transonic wings, turbines

- Where the flow over the suction side of a transonic wing is accelerated to a supersonic speed, the resulting re-compression can be by either Prandtl-Meyer compression or by the formation of a normal shock. This shock is of particular interest to makers of transonic devices because it can cause separation of the boundary layer at the point where it touches the transonic profile. This can then lead to full separation and stall on the profile, higher drag, or shock-buffet, a condition where the separation and the shock interact in a resonance condition, causing resonating loads on the underlying structure.

Shock in a pipe flow

- This shock appears when supersonic flow in a pipe is decelerated.
- Examples: Supersonic ramjet, scramjet, needle valve
- In this case the gas ahead of the shock is supersonic (in the laboratory frame), and the gas behind the shock system is either supersonic (*oblique shock*s) or subsonic (a *normal shock*) (Although for some oblique shocks very close to the deflection angle limit, the downstream Mach number is subsonic.) The shock is the result of the deceleration of the gas by a converging duct, or by the growth of the boundary layer on the wall of a parallel duct.

Shock waves in rapid granular flows

Shock waves can also occur in rapid flows of dense granular materials down inclined channels or slopes. Strong shocks in rapid dense granular flows can be studied theoretically and analyzed to compare with experimental data. Consider a configuration in which the rapidly moving material down the chute impinges on an obstruction wall erected perpendicular at the end of a long and steep channel. Impact leads to a sudden change in the flow regime from a fast moving supercritical thin layer to a stagnant thick heap. This flow configuration is particularly interesting because it is analogous to some hydraulic and aerodynamic situations associated with flow regime changes from supercritical to subcritical flows. Such study is important in estimating impact pressures exerted by avalanches and granular flows on defense structures or infrastructure

along the channel and in the run-out zones, and to study the complex flow dynamics around the obstacles and in depositions when the mass comes suddenly to a standstill.

Shock waves in astrophysics

Main article: Shock waves in astrophysics

Astrophysical environments feature many different types of shock waves. Some common examples are supernovae shock waves or blast waves traveling through the interstellar medium, the bow shock caused by the Earth's magnetic field colliding with the solar wind and shock waves caused by galaxies colliding with each other. Another interesting type of shock in astrophysics is the quasi-steady reverse shock or termination shock that terminates the ultra relativistic wind from young pulsars.

UNIT V PROPULSION

Centuries ago in 100 A.D., Hero, a Greek philosopher and mathematician, demonstrated jet power in a machine called an "aeolipile." A heated, water filled steel ball with nozzles spun as steam escaped.

Over the course of the past half a century, jet-powered flight has vastly changed the way we all live. However, the basic principle of jet propulsion is neither new nor complicated.

Centuries ago in 100 A.D., Hero, a Greek philosopher and mathematician, demonstrated jet power in a machine called an "aeolipile." A heated, water filled steel ball with nozzles spun as steam escaped. Why? The principle behind this phenomenon was not fully understood until 1690 A.D. when Sir Isaac Newton in England formulated the principle of Hero's jet propulsion "aeolipile" in scientific terms. His Third Law of Motion stated: "Every action produces a reaction ... equal in force and opposite in direction."

The jet engine of today operates according to this same basic principle. Jet engines contain three common components: the **compressor**, the **combustor**, and the **turbine**. To this basic engine, other components may be added, including:

- A **nozzle** to recover and direct the gas energy and possibly divert the thrust for vertical takeoff and landing as well as changing direction of aircraft flight.
- An **afterburner**or **augmentor**, a long "tailpipe" behind the turbine into which additional fuel is sprayed and burned to provide additional thrust.
- A **thrust reverser**, which blocks the gas rushing toward the rear of the engine, thus forcing the gases forward to provide additional braking of aircraft.
- A **fan** in front of the compressor to increase thrust and reduce fuel consumption.

An additional **turbine** that can be utilized to drive a **propeller** or **helicopter rotor**.

Pressure and Velocity. Air is normally thought of in relation to its temperature, pressure, and volume. Within a gas turbine engine the air is put into motion so now another factor must be considered, velocity. Consider a constant airflow through a duct. As long as the duct crosssectional area remains unchanged, air will continue to flow at the same rate (disregard frictional loss). If the crosssectional area of the duct should become smaller (convergent)the airflow must increase velocity if it is to continue to flow the same number of pounds per second of airflow (Bernoulli's Principle). In order to obtain the necessary velocity energy to accomplish this, the air must give up some pressure and temperature energy (law of conservation of energy). The net result of flow through this restriction would be a decrease in pressure and temperature and an increase in velocity. The opposite would be true if air were to

flow from a smaller into a larger duct (divergent area); velocity would then decrease, and pressure and temperature would increase. The throat of an automobile carburetor is a good example of the effect of airflow through a restriction (venturi); even on the hottest day the center portion of the carburetor feels cool. Convergent and divergent areas are used throughout a gas turbine engine to control pressure and velocity of the airgas stream as it flows through the engine.

jet engine is a reaction engine that discharges a fast moving jet of fluid to generate thrust by *jet propulsion* and in accordance with Newton's laws of motion. This broad definition of jet engines includes turbojets, turbofans, rockets, ramjets, pulse jets and pump-jets. In general, most jet engines are internal combustion engines[1] but non-combusting forms also exist.

In common parlance, the term *jet engine* loosely refers to an internal combustion airbreathing jet engine (a *duct engine*). These typically consist of an engine with a rotary (rotating) air compressor powered by a turbine ("Brayton cycle"), with the leftover power providing thrust via a propelling nozzle. These types of jet engines are primarily used by jet aircraft for long distance travel. Early jet aircraft used turbojet engines which were relatively inefficient for subsonic flight. Modern subsonic jet aircraft usually use high-bypass turbofan engines which give high speeds, as well as (over long distances) better fuel efficiency than many other forms of transport.

History

Jet engines can be dated back to the invention of the aeolipile before the first century AD. This device used steam power directed through two nozzles to cause a sphere to spin rapidly on its axis. So far as is known, it was not used for supplying mechanical power, and the potential practical applications of this invention were not recognized. It was simply considered a curiosity.

Jet or rocket propulsion only took off, literally and figuratively, with the invention of the gunpowder-powered rocket by the Chinese in the 13th century as a type of fireworks, but gradually progressed to propel formidable weaponry; and there the technology stalled for hundreds of years.

The earliest attempts at jet engines were hybrid designs in which an external power source first compressed air, which was then mixed with fuel and burned for jet thrust. In one such system, called a *thermojet* by Secondo Campini but more commonly, motorjet, the air was compressed by a fan driven by a conventional piston engine. Examples of this type of design were the Caproni Campini N.1, and the Japanese Tsu-11 engine intended to power Ohka kamikaze planes towards the end of World War II. None were entirely successful and the N.1 ended up being slower than the same design with a traditional engine and propeller combination.

Even before the start of World War II, engineers were beginning to realize that the piston engine was self-limiting in terms of the maximum performance which could be attained; the limit was due to issues related to propeller efficiency,[2] which declined as blade tips approached the speed of sound. If engine, and thus aircraft, performance were ever to increase beyond such a barrier, a way would have to be found to radically improve the design of the piston engine, or a wholly new type of powerplant would have to be developed. This was the motivation behind the development of the gas turbine engine, commonly called a "jet" engine, which would become almost as revolutionary to aviation as the Wright brothers' first flight.

The key to a practical jet engine was the gas turbine, used to extract energy from the engine itself to drive the compressor. The gas turbine was not an idea developed in the 1930s: the patent for a stationary turbine was granted to John Barber in England in 1791. The first gas turbine to successfully run self-sustaining was built in 1903 by Norwegian engineer Ægidius Elling. Limitations in design and practical engineering and metallurgy prevented such engines reaching manufacture. The main problems were safety, reliability, weight and, especially, sustained operation.

The first patent for using a gas turbine to power an aircraft was filed in 1921 by Frenchman Maxime Guillaume.[3] His engine was an axial-flow turbojet. Alan Arnold Griffith published *An Aerodynamic Theory of Turbine Design* in 1926 leading to experimental work at the RAE.

Types of Rocket Engines:

rocket or **rocket vehicle** is a missile, spacecraft, aircraft or other vehicle which obtains thrust from a rocket engine. In all rockets, the exhaust is formed entirely from propellants carried within the rocket before use. Rocket engines work by action and reaction. Rocket engines push rockets forwards simply by throwing their exhaust backwards extremely fast.

Rockets for military and recreational uses date back to the 13th century.[2] Significant scientific, interplanetary and industrial use did not occur until the 20th century, when rocketry was the enabling technology of the Space Age, including setting foot on the moon.

Rockets are used for fireworks, weaponry, ejection seats, launch vehicles for artificial satellites, human spaceflight and exploration of other planets. While comparatively inefficient for low speed use, they are very lightweight and powerful, capable of generating large accelerations and of attaining extremely high speeds with reasonable efficiency.

Chemical rockets are the most common type of rocket and they typically create their exhaust by the combustion of rocket propellant. Chemical rockets store a large amount of energy in an easily released form, and can be very dangerous. However, careful design, testing, construction and use minimizes risks.

Rocket vehicles are often constructed in the archetypal tall thin "rocket" shape that takes off vertically, but there are actually many different types of rockets including:[58][59]

- tiny models such as balloon rockets, water rockets, skyrockets or small solid rockets that can be purchased at a hobby store
- missiles
- space rockets such as the enormous Saturn V used for the Apollo program
- rocket cars
- rocket bike
- rocket powered aircraft (including rocket assisted takeoff of conventional aircraft- JATO)
- rocket sleds
- rocket trains
- rocket torpedos
- rocket powered jet packs

- rapid escape systems such as ejection seats and launch escape systems
- space probes

Propellants: A **propellant** is a material that is used to move ("propel") an object. The material is usually expelled by gas pressure through a nozzle. The pressure may be from a compressed gas, or a gas produced by a chemical reaction. The exhaust material may be a gas, liquid, plasma, or, before the chemical reaction, a solid, liquid or gelled.Common chemical propellants consist of a fuel; like gasoline, jet fuel, rocket fuel, and an oxidizer.

Propellant used for propulsion

Technically, the word **propellant** is the general name for chemicals used to create thrust. For vehicles, the term propellant refers only to chemicals that are stored within the vehicle prior to use, and excludes atmospheric gas or other material that may be collected in operation.

Amongst the English-speaking laymen, used to having fuels propel vehicles on Earth, the word **fuel** is inappropriately[*dubious – discuss*] used. In Germany, the word *Treibstoff*—literally "drive-stuff"—is used; in France, the word *ergols* is used; it has the same Greek roots as hypergolic, a term used in English for propellants which combine spontaneously and do not have to be set ablaze by auxiliary ignition system.

In rockets, the most common combinations are *bipropellants*, which use two chemicals, a fuel and an oxidiser. There is the possibility of a tripropellant combination, which takes advantage of the ability of substances with smaller atoms to attain a greater exhaust velocity, and hence propulsive efficiency, at a given temperature.

Although not used in practice, the most developed tripropellant systems involves adding a third propellant tank containing liquid hydrogen to do this.

Solid propellant

In ballistics and pyrotechnics, a **propellant** is a generic name for chemicals used for propelling projectiles from guns and other firearms.

Propellants are usually made from low explosive materials, but may include high explosive chemical ingredients that are diluted and burned in a controlled way (deflagration) rather than detonation. The controlled burning of the propellant composition usually produces thrust by gas pressure and can accelerate a projectile, rocket, or other vehicle. In this sense, common or well known **propellants** include, for firearms, artillery and solid propellant rockets:

- Gun propellants, such as:
    - Gunpowder (black powder)
    - Nitrocellulose-based powders
    - Cordite
    - Ballistite
    - Smokeless powders
- Composite propellants made from a solid oxidizer such as ammonium perchlorate or ammonium nitrate, a rubber such as HTPB, or PBAN (may be replaced by energetic polymers such as polyglycidyl nitrate or polyvinyl nitrate for extra energy) , optional high explosive fuels (again, for extra energy) such as RDX or nitroglycerin, and usually a powdered metal fuel such as aluminum.
- Some amateur propellants use potassium nitrate, combined with sugar, epoxy, or other fuels / binder compounds.
- Potassium perchlorate has been used as an oxidizer, paired with asphalt, epoxy, and other binders.

Propellants that explode in operation are of little practical use currently, although there have been experiments with Pulse Detonation Engines.

Grain

Propellants are used in forms called grains. A grain is any individual particle of propellant regardless of the size or shape. The shape and size of a propellant grain determines the burn time, amount of gas and rate produced from the burning propellant and consequently thrust vs time profile.

There are three types of burns that can be achieved with different grains.

Progressive Burn

Usually a grain with multiple perforations or a star cut in the center providing a lot of surface area.

Digressive Burn

Usually a solid grain in the shape of a cylinder or sphere.

Neutral Burn

Usually a single perforation; as outside surface decreases the inside surface increases at the same rate.

Composition

There are four different types of solid propellant compositions:

Single Based Propellant: A single based propellant has nitrocellulose as its chief explosives ingredient. Stabilizers and other additives are used to control the chemical stability and enhance the propellant's properties.

Double Based Propellant: Double based propellants consist of nitrocellulose with nitroglycerin or other liquid organic nitrate explosives added. Stabilizers and other additives are used also. Nitroglycerin reduces smoke and increases the energy output. Double based propellants are used in small arms, cannons, mortars and rockets.

Triple Based Propellant

Triple based propellants consist of nitrocellulose, nitroquanidine, nitroglycerin or other liquid organic nitrate explosives. Triple based propellants are used in cannons.

Composite

Composites contain no nitrocellulose, nitroglycerin, nitroquanidine or any other organic nitrate. Composites usually consist of a fuel such as metallic aluminum, a binder such as synthetic rubber, and an oxidizer such as ammonium perchlorate. Composite propellants are used in large rocket motors.

Liquid propellant

Common propellant combinations used for liquid propellant rockets include:

- Red fuming nitric acid (RFNA) and kerosene or RP-1
- RFNA and Unsymmetrical dimethyl hydrazine (UDMH)
- Dinitrogen tetroxide and UDMH, MMH and/or hydrazine
- Liquid oxygen and kerosene or RP-1
- Liquid oxygen and liquid hydrogen
- Liquid oxygen and ethanol
- Hydrogen peroxide and alcohol or RP-1
- Chlorine pentafluoride and hydrazine

Common monopropellant used for liquid rocket engines include:

- Hydrogen peroxide
- Hydrazine
- Red fuming nitric acid (RFNA)

Introducing propellant into a combustion chamber

Rocket propellant is mass that is stored, usually in some form of propellant tank, prior to being ejected from a rocket engine in the form of a fluid jet to produce thrust.

Chemical rocket propellants are most commonly used, which undergo exothermic chemical reactions which produce hot gas which is used by a rocket for propulsive purposes. Alternatively, a chemically inert reaction mass can be heated using a high-energy power source via a heat exchanger, and then no combustion chamber is used.

A solid rocket motor:

Solid rocket propellants are prepared as a mixture of fuel and oxidizing components called 'grain' and the propellant storage casing effectively becomes the combustion chamber. Liquid-fueled rockets typically pump separate fuel and oxidiser components into the combustion chamber,

where they mix and burn. Hybrid rocket engines use a combination of solid and liquid or gaseous propellants. Both liquid and hybrid rockets use *injectors* to introduce the propellant into the chamber. These are often an array of simple jets- holes through which the propellant escapes under pressure; but sometimes may be more complex spray nozzles. When two or more propellants are injected the jets usually deliberately collide the propellants as this breaks up the flow into smaller droplets that burn more easily.

Rocket Ignition :

Rocket fuels, hypergolic or otherwise, must be mixed in the right quantities to have a controlled rate of production of hot gas. A hard start indicates that the quantity of combustible propellant that entered the combustion chamber prior to ignition was too large. The result is an excessive spike of pressure, possibly leading to structural failure or even an explosion (sometimes facetiously referred to as "spontaneous disassembly").

Avoiding hard starts involves careful timing of the ignition relative to valve timing or varying the mixture ratio so as to limit the maximum pressure that can occur or simply ensuring an adequate ignition source is present well prior to propellant entering the chamber.

Explosions from hard starts often cannot happen with purely gaseous propellants, since the amount of the gas present in the chamber is limited by the injector area relative to the throat area, and for practical designs propellant mass escapes too quickly to be an issue.

A famous example of a hard start was the explosion of Wernher von Braun's "1W" engine during a demonstration to General Dornberger on December 21, 1932. Delayed ignition allowed the chamber to fill with alcohol and liquid oxygen, which exploded violently. Shrapnel was embedded in the walls, but nobody was hit.

Rocket Combution:

Combustion chamber

For chemical rockets the combustion chamber is typically just a cylinder, and flame holders are rarely used. The dimensions of the cylinder are such that the propellant is able to combust

thoroughly; different propellants require different combustion chamber sizes for this to occur. This leads to a number called L^*:

$$L = V_c/A_t$$

where:

- V_c is the volume of the chamber
- A_t is the area of the throat

L* is typically in the range of 25–60 inches (0.63–1.5 m).

The combination of temperatures and pressures typically reached in a combustion chamber is usually extreme by any standards. Unlike in air-breathing jet engines, no atmospheric nitrogen is present to dilute and cool the combustion, and the temperature can reach true stoichiometric. This, in combination with the high pressures, means that the rate of heat conduction through the walls is very high.

Rocket nozzles

Typical temperatures (T) and pressures (p) and speeds (v) in a De Laval Nozzle

The large bell or cone shaped expansion nozzle gives a rocket engine its characteristic shape.

In rockets the hot gas produced in the combustion chamber is permitted to escape from the combustion chamber through an opening (the "throat"), within a high expansion-ratio 'de Laval' nozzle.

Provided sufficient pressure is provided to the nozzle (about 2.5-3x above ambient pressure) the nozzle *chokes* and a supersonic jet is formed, dramatically accelerating the gas, converting most of the thermal energy into kinetic energy.

The exhaust speeds vary, depending on the expansion ratio the nozzle is designed to give, but exhaust speeds as high as ten times the speed of sound of sea level air are not uncommon.

Rocket thrust is caused by pressures acting in the combustion chamber and nozzle. From Newton's third law, equal and opposite pressures act on the exhaust, and this accelerates it to high speeds.

About half of the rocket engine's thrust comes from the unbalanced pressures inside the combustion chamber and the rest comes from the pressures acting against the inside of the nozzle (see diagram). As the gas expands (adiabatically) the pressure against the nozzle's walls forces the rocket engine in one direction while accelerating the gas in the other.

Propellant efficiency

For a rocket engine to be propellant efficient, it is important that the maximum pressures possible be created on the walls of the chamber and nozzle by a specific amount of propellant; as this is the source of the thrust. This can be achieved by all of:

- heating the propellant to as high a temperature as possible (using a high energy fuel, containing hydrogen and carbon and sometimes metals such as aluminium, or even using nuclear energy)
- using a low specific density gas (as hydrogen rich as possible)
- using propellants which are, or decompose to, simple molecules with few degrees of freedom to maximise translational velocity

Since all of these things minimise the mass of the propellant used, and since pressure is proportional to the mass of propellant present to be accelerated as it pushes on the engine, and since from Newton's third law the pressure that acts on the engine also reciprocally acts on the propellant, it turns out that for any given engine the speed that the propellant leaves the chamber is unaffected by the chamber pressure (although the thrust is proportional). However, speed is significantly affected by all three of the above factors and the exhaust speed is an excellent measure of the engine propellant efficiency. This is termed *exhaust velocity*, and after allowance is made for factors that can reduce it, the **effective exhaust velocity** is one of the most important parameters of a rocket engine (although weight, cost, ease of manufacture etc. are usually also very important).

For aerodynamic reasons the flow goes sonic ("chokes") at the narrowest part of the nozzle, the 'throat'. Since the speed of sound in gases increases with the square root of temperature, the use of hot exhaust gas greatly improves performance. By comparison, at room temperature the speed of sound in air is about 340 m/s while the speed of sound in the hot gas of a rocket engine can be over 1700 m/s; much of this performance is due to the higher temperature, but additionally rocket propellants are chosen to be of low molecular mass, and this also gives a higher velocity compared to air.

Expansion in the rocket nozzle then further multiplies the speed, typically between 1.5 and 2 times, giving a highly collimated hypersonic exhaust jet. The speed increase of a rocket nozzle is mostly determined by its area expansion ratio—the ratio of the area of the throat to the area at the exit, but detailed properties of the gas are also important. Larger ratio nozzles are more massive but are able to extract more heat from the combustion gases, increasing the exhaust velocity.

Nozzle efficiency is affected by operation in the atmosphere because atmospheric pressure changes with altitude; but due to the supersonic speeds of the gas exiting from a rocket engine, the pressure of the jet may be either below or above ambient, and equilibrium between the two is not reached at all altitudes (See Diagram).

Back pressure and optimal expansion

For optimal performance the pressure of the gas at the end of the nozzle should just equal the ambient pressure: if the exhaust's pressure is lower than the ambient pressure, then the vehicle will be slowed by the difference in pressure between the top of the engine and the exit; on the other hand, if the exhaust's pressure is higher, then exhaust pressure that could have been converted into thrust is not converted, and energy is wasted.

To maintain this ideal of equality between the exhaust's exit pressure and the ambient pressure, the diameter of the nozzle would need to increase with altitude, giving the pressure a longer nozzle to act on (and reducing the exit pressure and temperature). This increase is difficult to arrange in a lightweight fashion, although is routinely done with other forms of jet engines. In rocketry a lightweight compromise nozzle is generally used and some reduction in atmospheric performance occurs when used at other than the 'design altitude' or when throttled. To improve

on this, various exotic nozzle designs such as the plug nozzle, stepped nozzles, the expanding nozzle and the aerospike have been proposed, each providing some way to adapt to changing ambient air pressure and each allowing the gas to expand further against the nozzle, giving extra thrust at higher altitudes.

When exhausting into a sufficiently low ambient pressure (vacuum) several issues arise. One is the sheer weight of the nozzle- beyond a certain point, for a particular vehicle, the extra weight of the nozzle outweighs any performance gained. Secondly, as the exhaust gases adiabatically expand within the nozzle they cool, and eventually some of the chemicals can freeze, producing 'snow' within the jet. This causes instabilities in the jet and must be avoided.

On a De Laval nozzle, exhaust gas flow detachment will occur in a grossly over-expanded nozzle. As the detachment point will not be uniform around the axis of the engine, a side force may be imparted to the engine. This side force may change over time and result in control problems with the launch vehicle.

Thrust vectoring

Many engines require the overall thrust to change direction over the length of the burn. A number of different ways to achieve this have been flown:

- The entire engine is mounted on a hinge or gimbal and any propellant feeds reach the engine via low pressure flexible pipes or rotary couplings.
- Just the combustion chamber and nozzle is gimbled, the pumps are fixed, and high pressure feeds attach to the engine
- multiple engines (often canted at slight angles) are deployed but throttled to give the overall vector that is required, giving only a very small penalty
- fixed engines with vernier thrusters
- high temperature vanes held in the exhaust that can be tilted to deflect the jet

Overall rocket engine performance

Rocket technology can combine very high thrust (meganewtons), very high exhaust speeds (around 10 times the speed of sound in air at sea level) and very high thrust/weight ratios (>100) *simultaneously* as well as being able to operate outside the atmosphere, and while permitting the use of low pressure and hence lightweight tanks and structure.

Rockets can be further optimised to even more extreme performance along one or more of these axes at the expense of the others.

Specific impulse

The most important metric for the efficiency of a rocket engine is impulse per unit of propellant, this is called specific impulse (usually written I_{sp}). This is either measured as a speed (the *effective exhaust velocity* V_e in metres/second or ft/s) or as a time (seconds). An engine that gives a large specific impulse is normally highly desirable.

The specific impulse that can be achieved is primarily a function of the propellant mix (and ultimately would limit the specific impulse), but practical limits on chamber pressures and the nozzle expansion ratios reduce the performance that can be achieved.

Space flight:

Spaceflight is the act of travelling into or through outer space. Spaceflight can occur with spacecraft which may, or may not, have humans on board. Examples of human spaceflight include the Russian Soyuz program, the U.S. Space shuttle program, as well as the ongoing International Space Station. Examples of unmanned spaceflight include space probes which leave Earth's orbit, as well as satellites in orbit around Earth, such as communication satellites.

Spaceflight is used in space exploration, and also in commercial activities like space tourism and satellite telecommunications. Additional non-commercial uses of spaceflight include space observatories, reconnaissance satellites and other earth observation satellites.

A spaceflight typically begins with a rocket launch, which provides the initial thrust to overcome the force of gravity and propels the spacecraft from the surface of the Earth. Once in space, the motion of a spacecraft—both when unpropelled and when under propulsion—is covered by the area of study called astrodynamics. Some spacecraft remain in space indefinitely, some disintegrate during atmospheric reentry, and others reach a planetary or lunar surface for landing or impact.

Types of spaceflight

Human spaceflight

The first human spaceflight was Vostok 1 on April 12, 1961, on which cosmonaut Yuri Gagarin of the USSR made one orbit around the Earth. In official Soviet documents, there is no mention of the fact that Gagarin parachuted the final seven miles.[3] The international rules for aviation records stated that "The pilot remains in his craft from launch to landing". This rule, if applied, would have "disqualified" Gagarins space-flight. Currently the only spacecraft regularly used for human spaceflight are Russian Soyuz spacecraft and the U.S. Space Shuttle fleet. Each of those space programs have used other spacecraft in the past. Recently, the Chinese Shenzhou spacecraft has been used three times for human spaceflight, and SpaceshipOne twice.

Sub-orbital spaceflight

On a sub-orbital spaceflight the spacecraft reaches space and then returns to the atmosphere after following a (primarily) ballistic trajectory. This is usually because of insufficient specific orbital energy, in which case a suborbital flight will last only a few minutes, but it is also possible for an object with enough energy for an orbit to have a trajectory that intersects the Earth's atmosphere, sometimes after many hours. Pioneer 1 was NASA's first space probe intended to reach the Moon. A partial failure caused it to instead follow a suborbital trajectory to an altitude of 113,854 kilometers (70,746 mi) before reentering the Earth's atmosphere 43 hours after launch.

The most generally recognized boundary of space is the Kármán line (actually a sphere) 100 km above sea level. (NASA alternatively defines an astronaut as someone who has flown more than 50 miles or 80 km above sea level.) It is not generally recognized by the public that the increase in potential energy required to pass the Kármán line is only about 3% of the orbital energy (potential plus kinetic energy) required by the lowest possible earth orbit (a circular orbit just above the Kármán line.) In other words, it is far easier to reach space than to stay there.

On May 17, 2004, Civilian Space eXploration Team launched the GoFast Rocket on a suborbital flight, the first amateur spaceflight. On June 21, 2004, SpaceShipOne was used for the first privately-funded human spaceflight.

Orbital spaceflight

A minimal orbital spaceflight requires much higher velocities than a minimal sub-orbital flight, and so it is technologically much more challenging to achieve. To achieve orbital spaceflight, the tangential velocity around the Earth is as important as altitude. In order to perform a stable and lasting flight in space, the spacecraft must reach the minimal orbital speed required for a closed orbit.

Interplanetary spaceflight

An artist's imaginative impression of a vehicle entering a wormhole for interstellar travel

Interplanetary travel is travel between planets within a single planetary system. In practice, the use of the term is confined to travel between the planets of the Solar System.

Interstellar spaceflight

Five spacecraft are currently leaving the Solar System on escape trajectories. The one farthest from the Sun is Voyager 1, which is more than 100 AU distant and is moving at 3.6 AU per year.[4] In comparison Proxima Centauri, the closest star other than the Sun, is 267,000 AU distant. It will take Voyager 1 over 74,000 years to reach this distance. Vehicle designs using other techniques, such as nuclear pulse propulsion are likely to be able to reach the nearest star significantly faster.

Another possibility that could allow for human interstellar spaceflight is to make use of time dilation, as this would make it possible for passengers in a fast-moving vehicle to travel further into the future while aging very little, in that their great speed slows down the rate of passage of on-board time. However, attaining such high speeds would still require the use of some new, advanced method of propulsion.

Intergalactic spaceflight

Intergalactic travel involves spaceflight between galaxies, and is considered much more technologically demanding than even interstellar travel and, by current engineering terms, is considered science fiction.

QUESTION BANK

PART-A (2 Marks)

UNIT-1

1) State the difference between compressible fluid and incompressible fluid ?
2) Define stagnation pressure?
3) Express the stagnation enthalpy in terms of static enthalpy and velocity of flow?
4) Explain Mach cone and Mach angle?
5) Define adiabatic process?
6) Define Mach number?
7) Define zone of action and zone of silence ?
8) Define closed and open system?
9) What is the difference between intensive and extensive properties?
10) Distinguish between Mach wave and normal shock?

UNIT-II

1) Differentiate Adiabatic and Isentropic process.
2) Differentiate nozzle and diffuser ?
3) What is Impulse function ?
4) Differentiate between adiabatic flow and adiabatic flow ?

5) State the expression for dA/A as a function of Mach number ?

6) Give the expression for T/To and T/T* for isentropic flow through variable area interms of Mach number ?

7) Draw the variation of Mach number along the length of a convergent divergent duct when it acts as a (a) Nozzle (b) Diffuser (c) Venturi

8) What is chocked flow through a nozzle?

9) What type of nozzle used for sonic flow and supersonic flow?

10) When does the maximum mass flow occur for an isentropic flow with variable area?

UNIT-III

1) What are the consumption made for fanno flow?

2) Differentiate Fanno flow and Rayleigh flow?

3) Explain chocking in Fanno flow?

4) Explain the difference between Fanno flow and Isothermal flow?

5) Write down the ratio of velocities between any two sections in terms of their Mach number in a fanno flow ?

6) Write down the ratio of density between any two section in terms of their Mach number in a fanno flow?

7) What are the three equation governing Fanno flow?

8) Give the expression to find increase in entropy for Fanno flow?

9) Give two practical examples where the Fanno flow occurs?

10) What is Rayleigh line and Fanno line?

UNIT-IV

1) What is mean by shock wave ?

2) What is mean by Normal shock?

3) What is oblique shock?

4) Define strength of shock wave?

5) What are applications of moving shock wave ?

6) Shock waves cannot develop in subsonic flow? Why?

7) Define compression and rarefaction shock? Is the latter possible?

8) State the necessary conditions for a normal shock to occur in compressible flow?

9) Give the difference between normal and oblique shock?

10) what are the properties change across a normal shock ?

UNIT-V

1) Differentiate jet propulsion and rocket propulsion (or) differentiate between air breating and rocket

propulsion?

2) What is monopropellant? Give one example for that?

3) What is bipropellant?

4) Classify the rocket engines based on sources of energy employed?

5) What is specify impulse of rocket?

6) Define specific consumption?

7) What is weight flow co-efficient?

8) What is IWR?

9) What is thrust co-efficient?

10) Define propulsive efficiency?

Part - B (16 Marks)

UNIT-1

1. Stating the assumptions used.An air jet (r =1.4, R=287 J/Kg K) at 400K has sonic velocity .Determine:

1. velocity of sound at 400 K

2. Velocity of sound at the stagnation conditions.

3. Maximum velocity of the jet.

4. Staganation enthalpy.

5. crocco number.

2) The pressure, temperature and Mach number at the entry of a flow passage are 2.45 bar, 26.5° C and 1.4 respectively. If the exit Mach number is 2.5 determine for adiabatic flow of perfect gas (_ =1.3, R=0.469 KJ/Kg K).

3) Air (_ =1.4,R=287.43 J/Kg K) enters a straight axis symmetric duct at 300 K,3.45 bar and 150 m/s and leaves it at 277 k,500cm² . Assuming adiabatic flow determines:

1. stagnation temperature, 2. maximum velocity,

3. mass flow rate, and,

4. area of cross-section at exit.

4) An aircraft flies at 800 Km/hr at an altitude of 10,000 meters (T=223.15 K,P=0.264 bar). The air is reversibly compressed in an inlet diffuser. If the Mach number at the exit of the diffuser is 0.36 determine (a) entry Mach number and (b) velocity, pressure and temperature of air at diffuser exit. (16)

5) Air (Cp =1.05 KJ/Kg K,_ =1.38) at p1 =3*10 ^5 N/m² and T1 =500 k flows with a velocity of 200 m/s in a 30 cm diameter duct. Calculate mass flow rate,stagnation temperature,Mach number, and,Stagnation pressure values assuming the flow as compressible and incompressible. (16)

6) (a) What is the effect of Mach number on compressibility prove for

_=1.4, Þo –þ / ½ þ c² = 1 +¼ M² + 1/40 M 4 + ……. (8)

(b) Show that for sonic flow the deviation between the compressible and incompressible flow values of the pressure coefficient of a percent gas (_ =1.4) is about 27.5 per cent. (8)

7) Air at stagnation condition has a temperature of 800 K. Determine the stagnation velocity of Sound and the maximum possible fluid velocity. What is the velocity of the sound when the flow velocity is at half the maximum velocity (16)

8) Air flow through a duct. The pressure and temperature at station one are pressure is0 .7 bar and temperature is 300C. At a second station the pressure is 0.5 bar. Calculate temperature and density at the second station. Assume the flow is to be Isentropic (16)

Unit -II

1)Air flowing in a duct has a velocity of 300 m/s ,pressure 1.0 bar and temperature 290 k. Taking _=1.4 and R =287J/Kg K determine:

1) Stagnation pressure and temperature, (4)

2) Velocity of sound in the dynamic and stagnation conditions, (6)

3) Stagnation pressure assuming constant density. (6)

2) A conical diffuser has entry and exit diameters of 15 cm and 30cm respectively. .The pressure ,temperature and velocity of air at entry are 0.69bar,340 k and 180 m/s respectively . Determine

1) The exit pressure, (4)

2) The exit velocity and (6)

3) The force exerted on the diffuser walls. (6)

Assume isentropic flow,_ =1.4,Cp =1.00 KJ Kg-K.

3) A nozzle in a wind tunnel gives a test –section Mach number of 2.0 .Air enters the nozzle from a large reservoir at 0.69 bar and 310 k .The cross –sectional area of the throat is 1000cm².Determine the following quantites for the tunnel for one dimensional isentropic flow

1) Pressures,temperature and velocities at the throat and test sections, (4)

2) Area of cross- sectional of the test section , (4)

3) Mass flow rate, (4)

4) Power rate required to drive the compressor. (4)

4) Air is discharged from a reservoir at Po =6.91bar and To =325°c through a nozzle to an exit pressure of 0.98 bar .If the flow rate is 3600Kg/hr determine for isentropic flow:

1)Throat area, pressure,and velocity, (6)

2)Exit area,Mach number ,and (6)

3)Maximum velocity. (4)

5) A super sonic wind tunnel settling chamber expands air or Freon-21 through a nozzle from a nozzle from a pressure of 10 bar to 4bar in the test section . calculate the stagnation temperature to the maintained in the setting chamber to obtain a velocity of 500 m/s in the test section for Air ,Cp =1.025 KJ/Kg K, Cv =0.735 KJ/Kg K, Freon -21 , Cp =0.785 KJ/Kg K ,Cv= 0.675 KJ/Kg K.

What is the test section Mach number in each case ? (16)

6) Derive the following relations for one dimensional isentropic flow:

_ $dA/A = dP/\rho c^2(1 - M^2)$ (8)

_ $p^*/p = (2/_+1 + _-1 /_+1M^2)$ (8)

7) Air flowing in a duct has a velocity of 300 m/s ,pressure 1.0 bar and temperature 290 k.

Taking _=1.4 and R =287J/Kg K determine:

1)Stagnation pressure and temperature, (4)

2)Velocity of sound in the dynamic and stagnation conditions (6)

3)Stagnation pressure assuming constant density. (6)

8) A conical diffuser has entry and exit diameters of 15 cm and 30cm respectively

The pressure ,temperature and velocity of air at entry are 0.69bar,340 k and

180 m/s respectively . Determine

1) The exit pressure, (4)

2)The exit velocity and (6)

3)The force exerted on the diffuser walls. (6)

Assume isentropic flow,_ =1.4,Cp =1.00 KJ Kg-K.

9) A nozzle in a wind tunnel gives a test –section Mach number of 2.0 .Air enters the nozzle from a large reservoir at 0.69 bar and 310 k .The cross –sectional area of the throat is 1000cm².

Determine the following quantites for the tunnel for one dimensional isentropic flow:

1)Pressures,temperature and velocities at the throat and test sections, (4)

2)Area of cross- sectional of the test section , (4)

3)Mass flow rate, (4)

4)Power rate required to drive the compressor. (4)

10) Air is discharged from a reservoir at Po =6.91bar and To =325°c through a nozzle to an exit pressure of 0.98 bar .If the flow rate is 3600Kg/hr determine for isentropic flow:

1)Throat area, pressure,and velocity, (6)

2)Exit area,Mach number ,and (6)

3)Maximum velocity. (4)

11) A super sonic wind tunnel settling chamber expands air or Freon-21 through a nozzle from a nozzle from a pressure of 10 bar to 4bar in the test section . calculate the stagnation temperature to the maintained in the setting chamber to obtain a velocity of 500 m/s in the test section for, Air ,Cp =1.025 KJ/Kg K, Cv =0.735 KJ/Kg K, Freon -21 ,Cp =0.785 KJ/Kg K Cv= 0.675 KJ/Kg K.

What is the test section Mach number is each case ? (16)

12) Derive the following relations for one dimensional isentropic flow:

_ $dA/A = dP/\rho c^2(1 - M^2)$ (8)

_ $p^*/p = (2/_+1 + _-1 /_+1M^2)$ (8)

Unit -III

1)A circular duct passes 8.25Kg/s of air at an exit Mach number of 0.5. The entry pressure and temperature are 3.45 bar and 38°C respectively and the coefficient of friction 0.005.If the Mach number at entry is 0.15, determine :

I. The diameter of the duct , (2)

II. Length of the duct, (4)

III. Pressure and temperature at the exit, (4)

IV. Stagnation pressure loss, and (4)

V. Verify the exit Mach number through exit velocity and temperature. (2)

2) A gas (_ =1.3,R=0.287 KJ/KgK) at p1 =1bar, T1 =400 k enters a 30cm diameter duct at a Mach number of 2.0.A normal shock occurs at a Mach number of 1.5 and the exit Mach number is1.0,If the mean value of the friction factor is 0.003 determine:

1)Lengths of the duct upstream and downstream of the shock wave, (6)

2)Mass flow rate of the gas and (4)

3)Change of entropy upstream and downstream of the shock, across the shock and downstream of the shock. (6)

3) Air enters a long circular duct (d =12.5cm,f=0.0045) at a Mach number 0.5, pressure 3.0 bar and temperature 312 K.If the flow is isothermal throughout the duct determine (a) the length of the duct required to change the Mach number to 0.7,(b) pressure and temperature of air at M =0.7 (c) the lengthof the duct required to attain limiting Mach number, and (d) state of air at the limiting Mach number.compare these values with those obtained in adiabatic flow. (16)

4) A convergent –divergent nozzle is provided with a pipe of constant cross-section at its exit the exit diameter of the nozzle and that of the pipe is 40cm. The mean coefficient of friction for the pipe is 0.0025. Stagnation pressure and temperature of air at the nozzle entry are 12 bar and 600k. The flow is isentropic in the nozzle and adiabatic in the pipe.The Mach numbers at the entry and exit of the pipe are 1.8 and 1.0 respectively . Determine

a) The length of the pipe , (4)

b) Diameter of the nozzle throat,and (6)

c) Pressure and temperature at the pipe exit. (6)

5) Show that the upper and lower branches of a Fanno curve represent subsonic and supersonic flows respectively . prove that at the maximum entropy point Mach number is unity and all processes approach this point .How would the state of a gas in a flow change from the supersonic to subsonic branch ? (16)

Flow in constant area ducts with heat transfer(Rayleigh flow)

6) The Mach number at the exit of a combustion chamber is 0.9. The ratio of stagnation temper ature at exit and entry is 3.74. If the pressure and temperature of the gas at exit are 2.5 bar and 1000°C respectively determine (a) Mach number ,pressure and temperature of the gas at entry ,(b) the heat supplied per kg of the gas and (c) the maximum heat that can be supplied. Take _= 1.3, Cp= 1.218 KJ/KgK (16)

7) The conditions of a gas in a combuster at entry are: P1=0.343bar ,T1 = 310K ,C1= 60m/s. Detemine the Mach number ,pressure ,temperature and velocity at the exit if the increase in stagnation enthalpy of the gas between entry and exit is 1172.5KJ/Kg.

Take Cp=1.005KJ/KgK, _ =1.4 (16)

8) A combustion chamber in a gas turbine plant receives air at 350 K ,0.55bar and 75 m/s .The air –

fuel ratio is 29 and the calorific value of the fuel is 41.87 MJ/Kg .Taking _=1.4 and R =0.287 KJ/kg K for

the gas determine.

a) The initial and final Mach numbers, (4)

b) Final pressure ,temperature and velocity of the gas, (4)

c) Percent stagnation pressure loss in the combustion chamber , and (4)

d) The maximum stagnation temperature attainable. (4)

9) Obtain an equation representing the Rayleigh line . Draw Rayleigh lines on the h-s and p-v planes for two different values of the mass flux. Show that the slope of the Rayleigh line on the p-v plane is $\{dp/dv\} = \rho^2 c^2$ (16)

Unit-IV

Flow with normal shock

1)The state of a gas (_=1.3,R =0.469 KJ/Kg K) upstream of a normal shock is given by the following data:

Mx =2.5, px= 2bar ,Tx =275K calculate the Mach number ,pressure,temperature and velocity of the gas downstream of the shock; check the calculated values with those give in the gas tables. (16)

2) The ratio of th exit to entry area in a subsonic diffuser is 4.0 .The Mach number of a jet of air approaching the diffuser at p0=1.013 bar, T =290 K is 2.2 .There is a standing normal

shock wave just outside the diffuser entry. The flow in the diffuser is isentropic . Determine at the exit of the diffuser.

1. Mach number , (4)

2. Temperature, and (4)

3. Pressure (4)

4. What is the stagnation pressure loss between the initial and final states of the flow ? (4)

3) The velocity of a normal shock wave moving into stagnant air (p=1.0 bar, t=17°C) is 500 m/s .

If the area of cross- section of the duct is constant determine (a) pressure (b) temperature (c) velocity of air (d) stagnation temperature and (e) the mach number imparted upstream of the wave front. (16)

4) The following data refers to a supersonic wind tunnel:

Nozzle throat area =200cm²

Test section cross- section =337.5cm²

Working fluid ;air (_ =1.4, Cp =0.287 KJ/Kg K)

Determine the test section Mach number and the diffuser throat area if a normal shock is located in the test section. (16)

5) A supersonic diffuser for air (_ =1.4) has an area ratio of 0.416 with an inlet Mach number of 2.4 (design value). Determine the exit Mach number and the design value of the pressure ratio across the diffuser for isentropic flow. At an off- design value of the inlet Mach number (2.7) a normal shock occurs inside the diffuser .Determine the upstream Mach number and area ratio at the section where the shock occurs, diffuser efficiency and the pressure ratio across the diffuser. Depict graphically the static pressure distribution at off design. (16)

6) Starting from the energy equation for flow through a normal shock obtain the following relations (or) prandtl – meyer relation Cx Cy =a* ² M*x M*y =1 (16)

Flow with oblique shock waves:

7) Air approaches a symmetrical wedge (_ =15°) at a Mach number of 2.0.Determine for the strong and weak waves (a) wave angle (b) pressure ratio (c) density ratio, (d) temperature ratio and (e)downstream Mach number Verify these values using Gas tables for normal shocks. (16)

8) A gas (_ =1.3) at p1 =345 Mbar, T1= 350 K and M1=1.5 is to be isentropically expanded to 138 Mbar. Determine (a) the deflection angle ,(b) final Mach number and (c) the temperature of the gas. (16)

9) A jet of air at Mach number of 2.5 is deflected inwards at the corner of a curved wall..The wave angle at the corner is 60°.Determine the deflection angle of the wall, pressure and temperature ratios and final Mach number. (16)

10) Derive the Rankine –Hugoniot relation for an oblique shock

Þ2 /þ 1 = _ +1 p2 _ +1 p2

-------- --- +1 -------- + ------

_ -1 p1 _ -1 p1

Compare graphically the variation of density ratio with the intial Mach number in isentropic flow and flow with oblique shock. (16)

11) The Mach number at the exit of a combustion chamber is 0.9. The ratio of stagnation temperature at exit and entry is 3.74.If the pressure and temperature of a gas at exit are 2.5 bar and 1000°C respectively determine (a) Mach number ,pressure and temperature of the gas at entry,(b) the heat supplied per Kg of the gas and (c) the maximum heat that can be supplied.

Take _ =1.3 and Cp =1.218 KJ/Kg K (16)

12) The conditions of a gas in a combuster at entry are: P1=0.343 bar,T1= 310K ,C1=60m/s Determine the Mach number ,pressure,temperature and velocity at the exit if the increase in stagnation enthalpy of the gas between entry and exit is 1172.5KJ/Kg.

Take Cp=1.005KJ/kg, _ =1.4. (16)

13) A combustion chamber in a gas turbine plant receives air at 350 K , 0.55 bar and 75m/s. The air –fuel ratio is 29 and the calorific value of the fuel is 41.87 MJ/Kg.

Taking _ =1.4 and R =0.287 KJ/Kg K for the gas determine :

a) The initial and final Mach number, (4)

b) Final pressure,temperature and velocity of the gas, (4)

c) Percent stagnation pressure loss in the combustion chamber and (4),

d) The maximum stagnation temperature attainable. (4)

14) Obtain an equation representing the rayleigh line. Draw Rayleigh lines on the h-s and p-v planes for two different values of the mass flux.

Show that the slope of the Rayleigh line on the p-v plane is {dP/dV} r = þ² c² (16)

Unit -V

1) A turboprop engine operates at an altitude of 3000 meters above mean sea level and an aircraft speed of 525 Kmph. The data for the engine is given below

Inlet diffuser efficience =0.875

Compressor efficieny =0.790

Velocity of air at compressor entry =90m/s

Properties of air :_ =1.4, Cp =1.005 KJ/kg K (16)

2) The diameter of the propeller of an aircraft is 2.5m; It flies at a speed of 500Kmph at an altitude of 8000m. For a flight to jet speed ratio of 0.75 determine (a) the flow rate of air through the propeller, (b) thrust produced (c) specific thrust, (d) specific impulse and (e) the thrust power. (16)

3) An aircraft flies at 960Kmph. One of its turbojet engines takes in 40 kg/s of air and expands the gases to the ambient pressure .The air –fuel ratio is 50 and the lower calorific value of the fuel is 43 MJ/Kg .For maximum thrust power determine (a)jet velocity (b) thrust (c) specific thrust (d) thrust power (e) propulsive, thermal and overall efficiencies and (f) TSFC (16)

3) A turbo jet engine propels an aircraft at a Mach number of 0.8 in level flight at an altitude of 10 km

The data for the engine is given below:

Stagnation temperature at the turbine inlet =1200K

Stagnation temperature rise through the compressor =175 K

Calorific value of the fuel =43 MJ/Kg

Compressor efficiency =0.75

Combustion chamber efficiency =0.975

Turbine efficiency =0.81

Mechanical efficiency of the power transmission between turbine and compressor =0.98

Exhaust nozzle efficiency=0.97

Specific impulse =25 seconds

Assuming the same properties for air and combustion gases calculate

_ Fuel –air ratio, (2)

_ Compressor pressure ratio, (4)

_ Turbine pressure ratio, (4)

_ Exhaust nozzles pressure ratio ,and (4)

_ Mach number of exhaust jet (2)

5) A ramjet engine operates at M=1.5 at an altitude of 6500m.The diameter of the inlet diffuser at

entry is 50cm and the stagnation temperature at the nozzle entry is 1600K.The calorific value of the fuel used is 40MJ/Kg .The properties of the combustion gases are same as those of air (_ =1.4, R=287J/Kg K). The velocity of air at the diffuser exit is negligible

Calculate

(a) the efficiency of the ideal cycle, (b) flight speed (c) air flow rate (d) diffuser pressure ratio

(e) fuel –ratio (f)nozzle pressure ratio (g) nozzle jet Mach number (h) propulsive efficiency

(i) and thrust. Assume the following values:D =0.90,B =0.98, j= 0.96.

Stagnation pressure loss in the combustion chamber =0.002Po2. (16)

7) A rocket flies at 10,080 Kmph with an effective exhaust jet velocity of 1400m/s and propellant

flow rate of 5.0Kg/s .If the heat of reaction of the propellants is 6500KJ/Kg of the

propel at mixture determine;

a) Propulsion efficiency and propulsion power, (6)

b) Engine output and thermal efficiency ,and (6)

c) Overall efficiency. (4)

7) Determine the maximum velocity of a rocket and the altitude attained from the following data:

Mass ratio =0.15

Burn out time =75s

Effective jet velocity =2500m/s

What are the values of the velocity and altitude losses due to gravity ?Ignore drag and

assume vertical trajectory . (16)

8) A missile has a maximum flight speed to jet speed ratio of 0.2105 and specific impulse

equal to 203.88 seconds .Determine for a burn out time of 8 seconds

a) Effective jet velocity (4)

b) Mass ratio and propellant mass functions (4)

c) Maximum flight speed, and (4)

d) Altitude gain during powered and coasting flights (4)

9) Calculate the orbital and escape velocities of a rocket at mean sea level and an altitude of 300km from the following data:

Radius of earth at mean sea level =6341.6Km

Acceleration due to gravity at mean sea level =9.809 m/s^2 (16)

10) With a neat sketches the principle of operation of:

1. turbo fan engine and (8)

2. ram jet engine (8)

11) Explain the construction and operation of a ramjet engine and derive an expression for the ideal efficiency. (16)

12) Explain the construction and operation of a solid propellant rocket engine. Also name any four solid propellants.and state its advantages and disadvantages. (16)

13) What are the advantages and disadvantages of liquid propellants compared to solid propellants.(16)

14) Dicuss in detail the various propellants used in solid fuel rockets and liquid fuel system .Also sketch the propellant feed-system for a liquid propellant rocket motor. (16)

15) Briefly explain the construction and working of :

A. Rocket engine (6)

B. Ramjet engine (6)

C. Pulsejet engine (4)

16) With the help of a neat sketch describe the working of a ramjet engine. Depict the various thermodynamic process occurring in it on h-s diagram. What is the effect of flight Mach number on its efficiency? (16)

17) Explain with a neat sketch the working of a turbo-pump feed system used in a liquid propellant rocket? (16)